CHAPTER 4

NUMERICAL SIMULATION

4.1 PURPOSE OF THE STUDY

The objective of the numerical simulation is to analyse the crashworthiness and axial collapse behaviour of conical frusta made out of aluminium, GFRP and the combination of both (i.e. Hybrid) materials and to compare the numerical results with the experimental results. In this regard, the commercially available ABAQUS 6.12 explicit finite element analysis (FEA) software was utilized. The techniques utilized for performing numerical simulation such as FE modelling, material model, interaction, contact, boundary conditions, FE simulation, and FE output are discussed in the following sections.

4.2 FE MODELING

The FE model consists of a conical frustum and two flat platens to simulate the actual crushing process. The conical frustum was modelled with actual dimensions and considered as a deformable body; whereas the flat platens were considered as rigid bodies. The models were meshed with the suitable elements, and the corresponding details are discussed in the subsequent sections.
4.2.1 Element Selection for Conical Frusta

The ABAQUS software has various types of elements and each element has its own variants as shown in Figures 4.1 (a) and (b). Based on the applications, the user can select the required type of elements and their alternative features. During crushing, the conical frusta were subjected to large deformation. To meet this requirement, the S4R and SC8R elements were found to be suitable for meshing the conical frusta models. The S4R and SC8R are 4-nodeed and 8-nodeed shell elements respectively. Both the types of elements are categorized as ‘3D continuum shell element’.

![Figure 4.1 (a) Various categories elements available in ABAQUS software and (b) the variants of shell element (ABAQUS-6.12 user manual)](image-url)
Since the SC8R element has more number of nodes in a single element, the accuracy of FE result was found to increase. However, it resulted in the increase of computation time unlike in S4R element. Further, the S4R element is also suitable for accommodating single or multi-layer material models and large strain analysis. Hence, the S4R element was found to be a better choice than SC8R element for this study. In order to create regular mesh pattern on conical frusta as shown in Figure 4.2 (a), the mapped mesh algorithm of ABAQUS was chosen. The angle-sectional behaviour of the shell was computed using the ‘Simpson’s’ thickness integration method, wherein three integration points were considered for each layer or lamina of the shell elements.

Figure 4.2  Meshed models of (a) conical frusta with S4R elements and (b) Rigid flat platen meshed with R3D4 elements

4.2.2  Element for Rigid Platens

There are two rigid platens of square size 160 mm x 160mm modelled separately and kept common for all the types of FE analysis. Rigid flat platens were meshed with 4-node 3D bilinear rigid quadrilateral element (R3D4) of size 4.6 mm. The coarse mesh was preferred for the rigid platen to solve contact issues between deformable body and rigid surface. Each rigid platen was assigned with individual rigid reference point as shown in
Figure 4.2 (b). All the loading and boundary conditions were assigned through the reference points of the respective rigid platens.

4.2.3 Laminate Ply Stock Modeling

In actual case, the aluminium GFRP and hybrid conical shell specimens were fabricated with different layers of materials. In FE modelling, in order to incorporate the actual case of conical frusta, the S4R element was subdivided into the required number of individual laminas. Each lamina or layer was assigned with the actual material properties and ply orientations as in the case of actual conical frusta. In the case of FE simulation of aluminium conical frusta, the aluminium shell thickness was considered as a single layer. Therefore, the single layer of S4R element was considered as a single lamina of aluminium material.

However, the GFRP conical frusta were fabricated with either 6 or 12 plies. Therefore, the single element was subdivided into either 6 or 12 sections as shown in Figure 4.3 (a). Each section was defined with unique ply name, material, thickness, co-ordinate system, rotating angle, and number of integration points.

For hybrid conical frustum, the single element was subdivided into the required number of individual plies, and each ply was assigned with material property, ply-orientation and thickness that correspond to the actual case of hybrid conical frusta. Hence, the single S4R element is composed of aluminium metal ply and the required number of GFRP plies with specific fibre-ply orientation and required thickness as shown in Figure 4.3 (b). The aluminium material property was assigned with the first ply and the rest of the plies were assigned with GFRP material properties. The cohesive layer in between adjacent laminates of GFRP was eliminated. However, the fracture
energy required for composite laminate damage is considered in this study to simulate the actual collapse of GFRP and hybrid specimens.

Figure 4.3  (a) Ply stock plot of GFRP conical frusta and (b) FE model and ply stock plot of hybrid conical frusta
4.3 MATERIAL MODEL

The aluminium, GFRP, or combination of both material properties were suitably assigned for various categories of FE models of conical frusta, and the same was considered as a deformable body. The corresponding details are discussed in the subsequent sections. The flat platens were considered as rigid bodies having infinity of stiffness. A known lumped mass was used for simulating impact loading condition, i.e. inertia effect was considered for dynamic analysis. In quasi-static loading, the inertia effect was neglected.

4.3.1 Aluminium Alloy

The aluminium alloy (AA6061) material properties were extracted from stress-strain plot given in Section 3.2.1. In ABAQUS, the behaviours of aluminium alloy material were represented as piecewise multi-linear elastic-plastic material which has mass density $\rho = 2700 \text{ kg/m}^3$, Young’s modulus $E = 70 \text{ GPa}$, Poisson’s ratio $\nu = 0.3$, yielding strength $\sigma_y = 55 \text{ MPa}$ and ultimate strength $\sigma_u = 101-105\text{MPa}$. For impact loading, the strain rate effect was assigned with aluminium alloy material model using Cowper-Symonds constitutive Equation (4.1), which relates the dynamic ($\sigma_{dy}$) and static yield ($\sigma_y$) stresses by the relation:

$$ \dot{\varepsilon} = D \left( \frac{\sigma_{dy}}{\sigma_y} - 1 \right)^n $$ (4.1)

Equation (4.1) is an overstress power law, which includes dynamic flow stress ($\sigma_{dy}$) at a uni-axial plastic strain rate ($\dot{\varepsilon}$) and static flow stress ($\sigma_y$). The constants $D$ and $n$ are the aluminium material parameters. The values for material parameters $D (= 6500 \text{ s}^{-1})$ and $n (= 4)$ were used for the FE simulation (Symonds 1965, Gupta et al. 2008a). The static yield stress was
modified according to the impact velocity by considering the piecewise linearity in the plastic region of the stress vs. strain curve. Equation (4.1) was also used to calculate the instantaneous values of the dynamic yield stress of aluminium material according to the static flow stress.

4.3.2 GFRP Material

The GFRP laminate material properties were obtained by testing the FRP laminates, and the same were used for GFRP material models of the conical frusta. Tables 3.3 and 3.4 show the mechanical properties and failure parameters of GFRP laminates respectively. For chopped ply oriented GFRP conical frusta, the quasi-orthotropic material model was assigned, which means that the Young’s modulus of FRP laminate is almost equal in all the directions. However, for woven [0/90] and angle-ply [±60°] oriented GFRP conical frusta, the fibre-ply orientation of each lamina was assigned by means of appropriate local coordinate system which corresponds to actual fibre-ply orientation in the conical frusta test specimens.

4.3.2.1 FRP lamina damage initiation and failure evolution

Due to the complexity of failure mechanisms in the GFRP and hybrid laminates, it is difficult to define an applicable failure criterion. In general, phenomenological strength criteria such as maximum stress, Hashin damage and Tsai-Wu criteria can be used to detect the failure status of composite laminates. In ABAQUS software, the fibre/matrix laminate damage initiation and failure evolution are used to predict using Hashin’s failure criterion. In the present study, the fibre laminate failures of GFRP and Hybrid specimen were predicted through Hashin’s failure model (Hashin & Rotem 1973, Hashin 1980). It includes determination of ‘damage initiation’ and ‘damage evolution’.
(i) **Damage Initiation**

The Hashin’s failure model considers four different damage initiation modes such as (a) fibre tension, (b) fibre compression, (c) matrix tension, and (d) matrix compression. The damage initiation refers to the onset of degradation at a material point. The damage initiation criteria for the fibre-reinforced composites in ABAQUS have the following general forms.

Fibre tension \((\sigma_{11} \geq 0)\)

\[
F^f_j = \left( \frac{\sigma_{11}}{X_T} \right)^2 + \alpha_C \left( \frac{\sigma_{12}}{S_L} \right)^2, \quad \sigma_{11} \geq 0
\tag{4.2}
\]

Fibre compression \((\sigma_{11} < 0)\)

\[
F^c_f = \left( \frac{\sigma_{11}}{X_C} \right)^2
\tag{4.3}
\]

Matrix tension \((\sigma_{22} \geq 0)\)

\[
F^t_m = \left( \frac{\sigma_{22}}{Y_T} \right)^2 + \left( \frac{\sigma_{12}}{S_L} \right)^2
\tag{4.4}
\]

Matrix compression \((\sigma_{22} < 0)\)

\[
F^c_m = \left( \frac{\sigma_{22}}{2S_T} \right)^2 + \left( \frac{Y_C}{2S_T} \right)^2 - 1 \left( \frac{\sigma_{22}}{Y_C} \right)^2 + \left( \frac{\sigma_{12}}{S_L} \right)^2
\tag{4.5}
\]

where \(X_T\) is longitudinal tensile strength, \(X_C\) - longitudinal compressive strength, \(Y_T\) - transverse tensile strength, \(Y_C\) - transverse compressive strength,
$S_L$ - longitudinal shear strength, $S_T$ - transverse shear strength, $\alpha_c$ - coefficient that determines the contribution of the shear stress to the fibre tensile initiation criterion and $\bar{\sigma}_{11}$ - longitudinal, $\bar{\sigma}_{22}$ - transverse and $\bar{\tau}_{12}$ - shear effective stress tensor components within the plane of the composite. The stress tensors are the components of effective stress tensor ($\sigma$). It is used for defining the damage initiation criteria. The GFRP laminate initiates the damage when any one failure parameter is equal to the value of unity, i.e. $F_{t}^{f} = 1, F_{m}^{t} = 1, F_{m}^{c} = 1$ or $F_{m}^{c} = 1$.

Prior to any damage initiation and the evolution of failures, the damage operator, $M$, is equal to the identity matrix, so $\bar{\sigma} = \sigma$ and it is in the form of Equation (4.6).

$$\bar{\sigma} = M \sigma$$ (4.6)

where $\sigma$ is the true stress, and $M$ is the damage operator. The $M$ is represented by Equation (4.7).

$$M = \begin{bmatrix} 1 \frac{1}{(1-d_f)} & 0 & 0 \\ 0 & \frac{1}{(1-d_m)} & 0 \\ 0 & 0 & \frac{1}{(1-d_s)} \end{bmatrix}$$ (4.7)

where $d_f$, $d_m$, and $d_s$ are internal damage variables. These variables are characterized by fibre, matrix, and shear damage, which are derived from damage variables $d_f^t, d_f^c, d_m^t$, and $d_m^c$ using Equation (4.2-4.5). The internal damage variables are represented in the following form as
Damage initiation occurs when one of the four aforementioned failure modes is satisfied, altering the corresponding damage parameters $d_f$, $d_m$ or $d_s$ so that the damage operator matrix is modified with new effective stress tensor values. The effective stress ($\sigma$) is intended to represent the stress acting over the damaged area that effectively resists the internal forces.

(ii) **Damage Evolution**

Once damage initiation and evolution have occurred for at least one mode, the damage operator becomes significant in the criteria for damage initiation of the other modes. In post-damage initiation model, the damage evolution response of the material is computed from Equation (4.11)

$$\sigma = C_d \varepsilon$$

where $\varepsilon$ is the strain, and $C_d$ is the damaged elasticity matrix which has the following form

$$C_d = \frac{1}{D_d} \begin{bmatrix} (1-d_f)E_1 & (1-d_f)(1-d_m)v_{21}E_1 & 0 \\ (1-d_f)(1-d_m)v_{12}E_2 & (1-d_m)E_2 & 0 \\ 0 & 0 & (1-d_s)GD_d \end{bmatrix}$$ (4.12)
where the parameter $D_d$ is the overall damage variable during damage evolution. It is represented as $D_d = 1 - (1 - df) (1 - dm) v_{12} v_{21}$. The parameters $df$ and $dm$ reflect the current state of fibre and matrix damage respectively. The parameter $ds$ reflect the current state of shear state. $G$ is the shear modulus. $E_1$ and $E_2$ are the Young's modulus of fibre laminates in longitudinal and transverse direction respectively. The $v_{12}$ and $v_{21}$ are the major and minor Poisson's ratios respectively.

Stress-strain relationships for damage are prone to mesh dependency during material softening, leading to erroneous results such as decreasing energy dissipation upon mesh refinement. A characteristic length, based on the element geometry and formulation, is introduced to improve the mesh dependency; so that the constitutive law is expressed as a stress-displacement ($\sigma$-$\delta$) relation (Lapczyk & Hurtado 2007). In this case, the damage variable will evolve in a bi-linear manner as shown in Figure 4.4 (a) for each of the four failure modes.

Figure 4.4 Damage evolution of fibre laminate (a) Equivalent stress-equivalent displacement and (b) Damage variable as a function of equivalent displacement.
The positive slope of the stress-displacement curve, line ‘OP’, prior to damage initiation represents linear elastic orthotropic behaviour. At point ‘P’ (i.e. equivalent displacement, $\delta_{eq}^0$, and stress, $\sigma_{eq}^0$, at the onset of damage), damage is initiated and evolves via degradation of material properties as indicated by the negative slope ‘PR’. Each increment is computed and stored so that unloading and re-loading of the partially damaged material can be accounted for as shown by line ‘OQ’. The energy dissipated due to failure, $G^c$, defines the equivalent displacement at final damage, $\delta_{eq}^f$ and is represented by the area under the triangle ‘OPR’. Hence,

$$
\delta_{eq}^f = \frac{2G^c}{\sigma_{eq}^0}
$$

(4.13)

After damage initiation (i.e., $\delta_{eq} \geq \delta_{eq}^0$), the damage variable for a particular mode of failure is given by the following expression, and this relation is also presented graphically in Figure 4.4 (b).

$$
d = \frac{\delta_{eq}^f(\delta_{eq} - \delta_{eq}^0)}{\delta_{eq}(\delta_{eq}^f - \delta_{eq}^0)}
$$

(4.14)

The fracture energy required for GFRP damage initiation and evolution ($G^c$) was obtained from literature (Lapczyk & Hurtado 2007 ) as $G^{c1} = 12.5$ N/mm and $G^{c2} = 1.0$ N/mm, which corresponds to the fracture energy of fibre in tension and compression respectively. The same was assigned to GFRP and hybrid specimens. Damage propagates when the total fracture energy in any of the four mentioned cases reaches its maximum value assigned by the user as an input parameter. After damage initiation, three independent non-negative in-ply damage parameters such as $d_f$, $d_m$, and $d_s$ reduce the ply stiffness numerically in fibre, transverse, and shear directions respectively, until the final failure point is reached.
4.4 CONTACT MODELING

Once the rigid flat platen and deformable conical frusta were developed separately, they were assembled in such a way that the conical frustum was sandwiched in between the rigid platens as shown in Figure 4.5. Since the platens were positioned on top and bottom ends of the conical frusta, they are named as rigid top platen and rigid bottom platen respectively. Further, the axis of the conical frustum was aligned in line with the centre of the rigid platens. A reliable contact modelling for the interaction between the conical frusta and top and bottom rigid platens is vital and needs to be accurately established.

![Assembled view of a FE model](image)

Figure 4.5 Assembled view of a FE model

To achieve reliable contact modelling, the surface of the moving top rigid platen and the outer surface of the conical frusta were established with surface-to-surface contact using the finite sliding ‘penalty’ based contact algorithm with contact pairs (Nagel & Thambiratnam 2006). Further, contact regions at the large end of the conical frusta and bottom rigid platen were related with ‘node-to-surface’ contact behaviour, and rough friction behaviour was considered over the contact boundaries. Since the friction coefficient is difficult to determine accurately under experimental conditions, it was treated
as a variable in the numerical model. From the trials of simulations, friction coefficients with the values of 0.35 and 0.2 were found to be suitable for the static and dynamic loading conditions respectively.

Self-contact interaction was also simulated using an ‘automatic single surface’ contact for the conical frusta in order to avoid interpenetration of the conical shell wall. An initial imperfection of conical frusta was not considered in numerical study. It was also evident that the numerical analyses of the conical frusta models without considering initial imperfections showed a good correlation with the experimental results. As pointed out by Reyes et al. (2002), introducing an initial imperfection only had a minor effect on the performance of energy-absorption.

4.5 BOUNDARY CONDITIONS

The actual boundary conditions were imposed on the FE model suitably in such a way that the movable rigid top platen was free to translate only along the ‘Y’ axis, and the rigid bottom supporting platen was fully constrained (Alghamdi et al. 2002b, Aljawi et al. 2005, Nagel & Thambiratnam 2006,) as shown in Figure 4.6.

![Figure 4.6 Schematic sketch of boundary conditions of a FE model](image)
4.6 LOADING CONDITIONS

The deformation rates were relatively small for an explicit solver; therefore, mass scaling was applied in order to achieve sufficiently small-time steps and to speed up the solution while still maintaining a quasi-static deformation condition. In this study, the quasi-static loading was simulated by performing mass scaling and prescribing a ramp velocity-time loading history.

Figure 4.7  Velocity-time history of the moving rigid body used in the (a) quasi-static and (b) impact simulations.

Figure 4.7 (a) shows the velocity-time history curve describing the motion of the rigid moving mass. The velocity was initially ramped up within ramping time, $t_R = 50$ ms (which provided acceptable results) and then followed by a constant velocity, $V$ of 12 mm/min throughout the total loading duration ($t_T$). In the case of impact loading, the initial impact velocity ($V_i$) of the crush plate just before the crushing is maximum. During crushing, the applied impact energy is absorbed by the specimen through material deformation, friction, and extrusion. Therefore, the final impact velocity of the crush plate becomes zero. Therefore, the higher level of initial impact velocity was assigned for the top rigid platen. Due to progressive loss of
crush-energy during crushing, the velocity-time profile was in the form of non-linear in trend as shown in Figure 4.7 (b).

According to the experimental loading conditions, the initial impact velocity and the ramping time were suitably selected and the same was assigned to the top rigid top platen. It should also be noted that the conical frustum is kept stationary over the bottom rigid platen, and the top platen is meant for crushing, but this condition is reverse in the case of experimental work. The conical specimen is subjected to impact over the stationary rigid platen. However, it would not affect the results (Nagel 2005) and the same was confirmed by simulating the FE model similar to the experimental test conditions.

4.7 FE SIMULATION

The rigid top platen is subjected to an axial compressive load over the small end of the conical frusta as shown in Figure 4.8. Since the quasi-static loading is to be performed in small increment of time intervals and the present FE model is having more nodes with material, geometry and contact non-linearity, the solution process consumes more computation time. In order to reduce the time steps, the mass densities of both aluminium and composite materials were scaled up to 1000 times of the original densities. The axial load of 100 N was applied along the Y-axis over the top end of the conical specimen through top rigid plate with a velocity of 12 mm/min. The top rigid plate was allowed to move downward only in ‘Y’ axis, whereas the bottom rigid plate was restrained to move in all directions. In ABAQUS explicit, scaling up the density and the relatively low loading velocity resulted in 0.2 mm/s of loading rate, which ensures a quasi-static simulation (El-Hage et al. 2007). In quasi-static simulation, the top rigid platen crushes the conical frusta up to the specified level of compression, i.e. displacement constraint movement.
In the case of impact loading, the axial impact velocity was applied along the Y-axis of the top rigid plate while the bottom platen was fixed. Different impact weights and velocities were considered for the simulation, and all the analyses were simulated in 0.02s of time step.

### 4.7.1 Selection of Mesh Size

The element choice and its size are the important factors, which determine the accuracy and the efficiency of the numerical analysis results. Therefore, trails of quasi-static FE simulations were performed with the developed aluminium conical FE models initially, and the effect of mesh size was studied in order to obtain the optimum mesh size for the conical FE model. In this regard, the FE model of the conical frusta was initially discretized with higher element size, and the simulations were performed. The corresponding mode of collapse and average crush load values were obtained. Similar to this case, the next set of simulations was performed by dividing the higher element size into half each time. On each trail, the collapse mode and average crush load values were monitored and the corresponding results are shown in Figures 4.9 (a) and (b).
From the figure, it was observed that the mesh size significantly influences the collapse pattern of FE conical model and the mean load. Further, it was observed that the collapse modes of FE model with 2 mm mesh size matches with the experimentally obtained results. Further, the reduction in mesh size resulted in more computation time without altering the accuracy of the results. The same was reflected in the mean collapse load value plot of AC15 model, wherein the mean load value is almost equal to 2 mm, 1.5 mm and 1.25 mm mesh sizes.

Hence, the mesh size of 2 mm was found to be optimum for this study. By keeping the mesh size as 2 mm, 6018–10455 elements were generated on various categories AC, GFRP and CWAC conical shell models. The other criteria of ABAQUS Explicit module were not altered. The artificial strain energy and the internal energy of each model were studied for
the possibility of solution convergence trends. It was also observed that the artificial energy should be less than 5% of internal energy of FE conical shell model, which resulted in good quality of meshing and quick convergence of solution (Mirfendereski et al. 2008, ABAQUS-6.12 manual).

4.8 FE OUTPUTS

The FE models with 2 mm mesh size were subjected to compression up to 30 mm from its original height. However, the crashworthiness of the conical frusta was investigated only up to 20 mm of axial compression, i.e. 20% sacrificing from its original height. The reaction force due to crush force of rigid top platen and its displacement was considered to determine load-deformation characteristics and the crashworthiness of conical frustum. With respect to the change of geometry and material and loading conditions, various types of collapse modes were predicted on AC, GFRP and hybrid (CWAC) models. For example, the progressive collapse behaviours of 18W3L model were predicted from FE analysis and it is shown in Figures 4.10 (a-g).

![Figure 4.10 (a-g) Progressive collapse of GFRP conical shell FE model (18W3L)](image)
The finite element analysis was carried out on various categories of conical frusta model by incorporating the mechanical properties of aluminium and GFRP materials. The quasi-static and impact loads were applied on the developed FE models. The collapse mode and the load-deformation plots were obtained. The FE plots were compared with the respective experimental results. The corresponding results are discussed in the next chapter.